International Journal of Mechanical Engineering and Technology (IJMET)
Volume 8, Issue 2, February 2017, pp. 237–245 Article ID: IJMET_08_02_029
Available online at http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=8&IType=2
ISSN Print: 0976-6340 and ISSN Online: 0976-6359

NUMERICAL SOLUTIONS FOR PERFORMANCE PREDICTION OF CENTRIFUGAL COMPRESSOR

G. Sravan Kumar
M Tech Student, Department of Mechanical Engineering,
AITAM Engineering College, Tekkali, Andhra Pradesh, India

Dr. D. Azad
Professor, Department of Mechanical Engineering,
AITAM Engineering College, Tekkali, Andhra Pradesh, India

K. Mohan Laxmi
Assistant Professor, Department of Mechanical Engineering,
AITAM Engineering College, Tekkali, Andhra Pradesh, India

K.K.K. Sanyasi Raju
Scientist-F (NSTL, Retd.), Visakhapatnam, Andhra Pradesh, India

ABSTRACT

An attempt is made in the present study to investigate the superior turbulence model for simulating three dimensional flows in centrifugal compressor. The strong channelled curvature and intensive rotations prevalent in centrifugal compressor resulting high swirling and secondary flow nictitates choosing appropriate turbulence model for accurate performance predictions. The various turbulence models offered in FLUENT viz Spalart Allmaras (curvature correction), Transition SST (curvature correction), Scaled Adaptive Simulations (Curvature correction with compressibility effect), Reynolds stress model (compressibility effect) were investigated presently for Eckardt Impeller. Reynolds stress model though involves higher computational time was found to be the superior model. It is essential to investigate the onset of surge and choke for completely understanding the performance of a centrifugal compressor. Choking phenomena was observed when the speed reached 16000 rpm with relative Mach number reaching unity in the impeller region. The maximum flow rate at 16000 rpm was 0.4 kg/s per blade and remained constant then 16500 rpm. Surging was founded to initiate when the back pressure has to reach 1.8 bar resulting in zero discharge.

Key words: Turbulence models; Choke; Surge; Complex flows; Strong channelled curvatures; Compressibility effects.

Cite this Article: G. Sravan Kumar, Dr. D. Azad, K. Mohan Laxmi and K.K.K. Sanyasi Raju. Numerical Solutions for Performance Prediction of Centrifugal Compressor. *International Journal of Mechanical Engineering and Technology*, 8(2), 2017, pp. 237–245.
http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=8&IType=2

1. INTRODUCTION

Centrifugal compressors are used in small gas turbines and are the driven units in most gas turbine compressors trains. They are integral part of petro chemical industry, finding extensive use because of their smooth operation, larger tolerance of process fluctuations and their highly reliability compared to other types of centrifugal compressors. Centrifugal compressors are used for high pressure ratios and lower flow rates compared to lower pressure ratios and higher flow rates in axial compressors. The most efficient region for the centrifugal compressor operation is in the specific speed range between $60< N_s<1500$. Specific speed of more than 3000 usually require axial flow compressor.

The CFD software provides a wide choice for choosing different turbulence models. However for turbo machinery, it is essential to use proper model for accurate prediction of performance due to strong channelled curvature and intensive rotation prevalent in a centrifugal compressor. Agheai (1) et.al during their investigation for superior turbulence model for centrifugal compressor of micro and radial turbines observed that the Renormalisation group model (RNG) was superior to k-ε standard and RSM models, the reason being that the results of RNG model showed more close agreement with experimental and one dimensional design data. Parvel (4) et.al tested the SST-CC model for predicting the total pressure rise of a centrifugal compressor at different design points. It was concluded by them that the SST-CC model results showed a considerably better agreement with the experimental values than the original SST model for most of the operating points and recommended SST-CC model due to its higher computational efficiency.

2. ONE DIMENSIONAL FLOW ANALYSIS

The impeller design usually starts with one dimensional flow analysis. The Eckardt impeller parameters (12) are obtained from literature survey and various other design parameters viz inlet flow velocity, absolute Mach number and relative Mach number etc. were computed and observed close agreement with the corresponding present3D CFD results. The basic impeller parameters and the computed values with corresponding CFD values are given below.

Table 1 geometric parameters of impeller.

Parameter	Unit	Specification
Blade numbers	-	20
Inlet radius at tip (r_s)	M	0.14
Inlet radius at hub (r_h)	M	0.045
Outlet radius (r_o)	M	0.2
Impeller exit width	M	0.026
Blade inlet angle (β_{1s})	deg	60
Blade back sweep (β_2)	deg	0

Table 2 One dimensional flow analysis of centrifugal compressor rotating at 14000 rpm.

Parameter	Analytical value	CFD value
Inlet Flow velocity (m/s)	98.18	95
Mass flow in (kg/s)	5.2	0.27/ Blade (RSM)
Inlet Mach number	0.29	0.32
Relative Mach number	0.67	0.689

3. NUMERICAL APPROACH

Various turbulence models investigated presently are given below

- Spalart-Allmaras (Curvature Correction).
- Transition SST (Curvature Correction).
- Reynolds stress model (Compressibility Effect).
- Scaled Adaptive Simulations (Curvature Correction +Compressibility Effect).

3.1. Grid Generation

The most important task in CFD process is mesh generation which takes a lot of time in the generation of the computational grid. First of all is the decision whether the domain shall be discretize with hexahedral, tetrahedral or hybrid cells. Using tetrahedral cells can save a lot of time because the meshing of the volume works almost automatically. In the present case, the mesh obtained is structural with 28000 hexahedral elements with 31311 nodes.

4. NUMERICAL IMPLEMENTATION

The simulation process in Fluent has two kinds of solvers

- Pressure-based solver
- Density-based coupled solver (DBCS)

In pressure-based solvers, momentum and pressure are considered as primary variables and by reformatting the continuity equation the Pressure-velocity coupling algorithms are derived, while in Density based coupled solver all equations regarding continuity, momentum, energy and species, if required are solved in vector form and solving pressure from equation of state. In the present case, as the flow is high speed compressible reaching the sonic value ,density based coupled solver is chosen in which Implicit solution approach is preferred to the explicit approach, due to its very strict limit on time step size and uses a point-implicit Gauss-Seidel / symmetric block Gauss-Seidel/ ILU method to solve for variables.

4.1. Boundary Conditions

For the compressible flow, pressure inlet condition was used as the inlet boundary condition. At the inlet, turbulent intensity and turbulent viscosity ratio must be defined too ($T_{\text{intensity}}$ = 10 percent, $T_{\text{viscosity ratio}}$ = 10). A pressure at the outlet and backflow properties must be defined. All reference frame moving walls are part of the rotating reference frame. These walls are treated as moving walls with rotational speed of 0 relative to the adjacent cell zone, which is rotating. Nonrotating walls in the inertia frame of reference that is part of the rotating reference frame are treated as moving walls with a rotational speed of absolute 0.The walls are adiabatic.

5. RESULTS AND DISCUSSION

5.1. Turbulence Models

The various performance parameters of compressor viz mass flow rate, relative Mach number, total pressure, axial force, torque, isentropic and polytrophic efficiencies are tabulated below for the various turbulence models investigated presently at constant speed of 14000 rpm and constant back pressure of 1.6 bar at the pressure outlet conditions.

It is observed from the table 3 that the results for all the turbulence models except for the mass flow rate are within 5%. However Spalart-Allmaras (CC) model is a low Reynolds number model and generally not recommended for turbo machinery problems. The transition SST model was considered as the best model for centrifugal compressor CFD problems (4) and Scaled Adaptive Simulations though giving same

values was not reported in the literature for the present problem. Out of all the models Reynolds stress model is considered more complex as it uses seven transport equations (six Reynolds stress components and one turbulent dissipation equation) which needs to be solved for each time step to provide Reynolds stress for the Navier-Strokes equations. Further the computational time for solving these seven equations would be higher when compared to the other turbulence models where the number of equations less than five. It is relevant to mention here that the Reynolds stress model is considered to be superior for the centrifugal compressors where the flow is very complex due to strong channelled flow and intensive rotations. In view of the above it is considered that the RSM model (CC) is the most suitable for centrifugal compressor related CFD problems.

Table 3 3D CFD calculations results at constant speed of 14000 rpm @ 1.6 bar back pressure.

Parameter	Turbulence model	CFD value
Mass flow (kg/s)	Spalart Allmaras(CC) Transition SST(CC) RSM(CE) SAS(CC+CE)	0.23 0.18 0.27 0.18
Relative Mach number	Spalart Allmaras(CC) Transition SST(CC) RSM(CE) SAS(CC+CE)	0.69 0.65 0.68 0.75
Relative velocity magnitude(m/s)	Spalart Allmaras(CC) Transition SST(CC) RSM(CE) SAS(CC+CE)	334 334 334 334
Mass average total pressure (bar)	Spalart Allmaras(CC) Transition SST(CC) RSM(CE) SAS(CC+CE)	2.22 2.15 2.20 2.12
Mass average total temperature (k)	Spalart Allmaras(CC) Transition SST(CC) RSM(CE) SAS(CC+CE)	364.45 360.89 366.12 363.90
Mass average radial flow angle (deg)	Spalart Allmaras(CC) Transition SST(CC) RSM(CE) SAS(CC+CE)	85.68 87.83 85.38 88.41
Mass average theta flow angle (deg)	Spalart Allmaras(CC) Transition SST(CC) RSM(CE) SAS(CC+CE)	88.28 89.35 87.83 89.49
Axial force (n)	Spalart Allmaras(CC) Transition SST(CC) RSM(CE) SAS(CC+CE)	1004.49 992.68 992.15 981.16
Torque (n-m)	Spalart Allmaras(CC) Transition SST(CC) RSM(CE) SAS(CC+CE)	12.93 10.11 14.85 10.47
Isentropic efficiency (%)	Spalart Allmaras(CC) Transition SST(CC) RSM(CE) SAS(CC+CE)	96.81 97.30 93.64 91.35

Polytrophic efficiency (%)	Spalart Allmaras(CC) Transition SST(CC) RSM(CE) SAS(CC+CE)	97.15 97.58 94.31 92.21

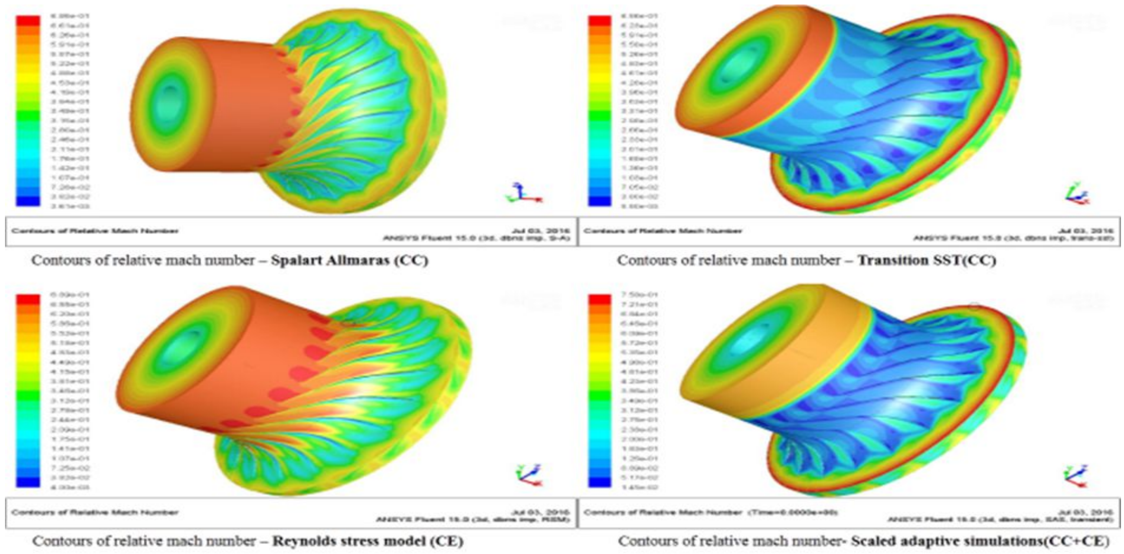


Figure 1 Contours of relative mach number for Spalart Allmaras, Transition SST, Reynolds stress model, Scaled Adaptive Simulation.

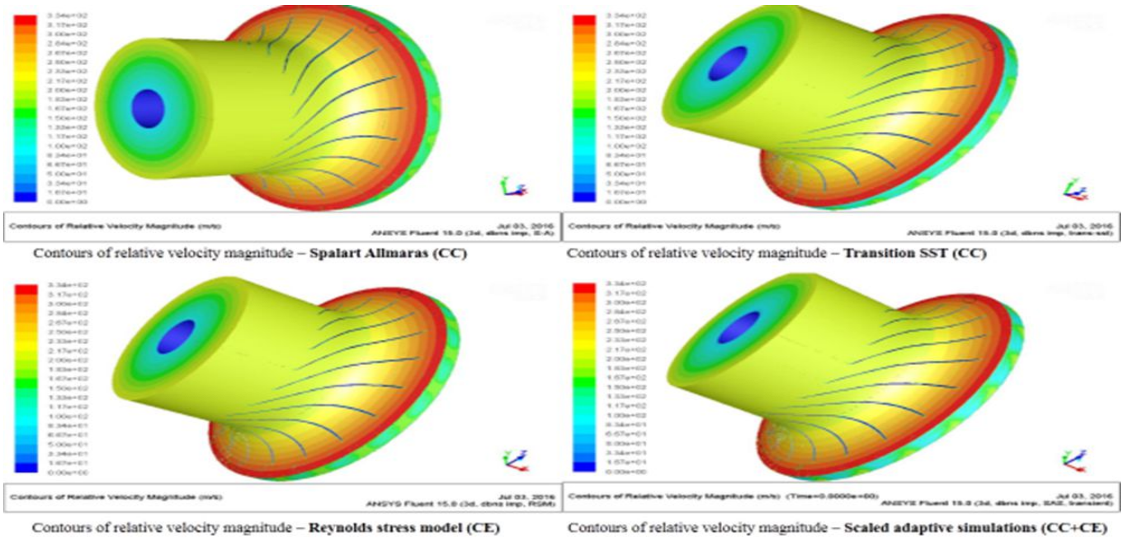


Figure 2 Contours of relative velocity magnitude for Spalart Allmaras, Transition SST, Reynolds stress model, Scaled Adaptive Simulation.

5.2. Surge Phenomena

A compressor is in "surge" when the main flow through the compressor reverses its direction and flows from the exit to the inlet for short time intervals. If allowed to persist these unsteady process may results in irreparable damage. Surge has been traditionally defined as the lower limit of stable operation in a compressor and involves the reversal of flow. Compressors are usually operated at a working line separated by some safety margin from the surge line. Surge is often symphonized by excessive vibration and an audible sound. However there has been cases in which surge problems that were not audible have causes failures.

It is possible to simulate surge phenomenon in Computational Fluid Dynamics by monitoring the changes in discharge with variation in back pressure. Variation of discharge with speed (at constant back pressure of 1.6 bars) and variation of discharge with pressure ratio at constant speed of 15000 rpm is presented below.

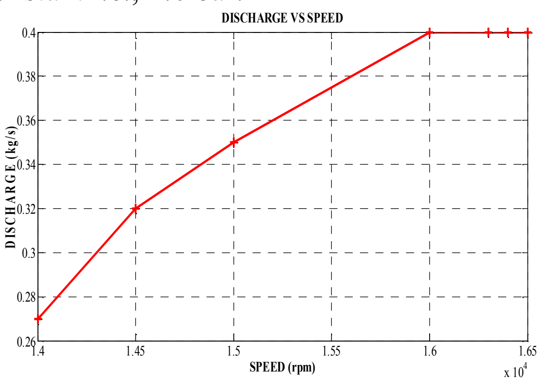
Table 4 Discharge, Pressure ratio vs Speed @1.6 bar back pressure.

S.no	Speed(rpm)	Mass flow rate (kg/s)	Pressure ratio (p_d/p_s)
1	14000	0.27	2.20
2	14500	0.32	2.29
3	15000	0.35	2.39
4	16000	0.4	2.64
5	16300	0.4	2.65
6	16400	0.4	2.65
7	16500	0.4	2.65

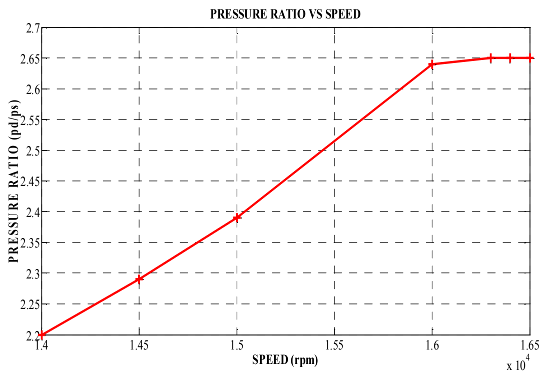
Table 5 Back pressure vs Discharge, Pressure ratio @ 15000 rpm.

S.no	Back pressure (bar)	Mass flow rate (kg/s)	Pressure ratio (p_d/p_s)
1	1.5	0.38	2.35
2	1.6	0.35	2.39
3	1.7	0.29	2.44
4	1.8	0	2.13

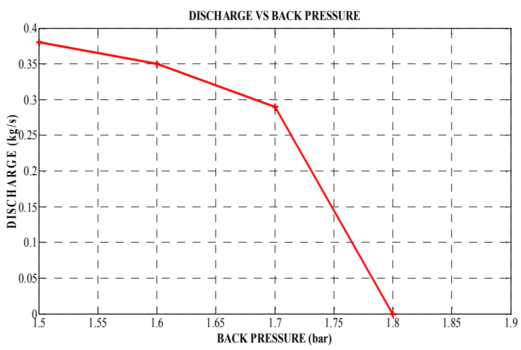
Referring to the tables 4 and the graphs 1&2, it is observed that discharge increases with increase in speed and remains constant after certain speed is attained. In the present case the discharge has reached maximum value of 0.4 kg/s at 16000 rpm and remains constant though the speed is increased from 16000 to 16500 rpm. However the above variation of discharge with speed is observed by maintaining the back pressure constant i.e., 1.6 bar.



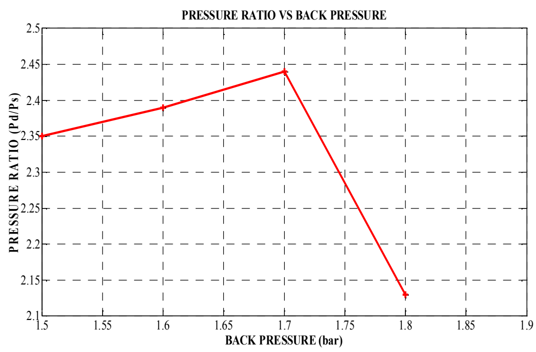
Graph 1 Discharge vs Speed.



Graph 2 Pressure ratio vs Speed.



Graph 3 Discharge vs Back pressure.



Graph 4 Pressure ratio vs Back pressure.

The pressure ratio also increases with speed as seen in graph 2. Table 5 refers to variation of discharge and Pressure ratio with back pressure, maintaining the same speed, 15000 rpm. It is seen that discharge almost reached zero value when the back pressure is increased to 1.8 bar whereas the pressure ratio also decreases indicating onset of surge phenomena which can be observed from graphs 3 and 4.

5.3. Choke Phenomena

Compressor choke is an abnormal operating condition for centrifugal compressor. As the flow rate increases, the discharge pressure decreases, i.e. resistance to flow are decreased and the reduction in density takes place. Beyond a certain limit, no possibility of increase in mass flow as we are reaching the right side of performance characteristics, this operating condition is also known as stonewalling of a centrifugal compressor. Choking takes place when the mass flow at any point reaches sonic velocity. Choking can occur either in the stationery part i.e. guide vanes or diffuser or in the rotating component When choking occurs the mass flow reaches maximum value and the relative velocity in the compressor part where choking is initiated reaches sonic velocity at a particular speed. In the present study the choke condition is simulated by varying the speed from 14000 to 16500 rpm.

Variation of discharge, pressure ratio and relative Mach number with speed are given below in table 6 and graphs 5. The contour plots of relative Mach number at different speeds is given at figure 3.

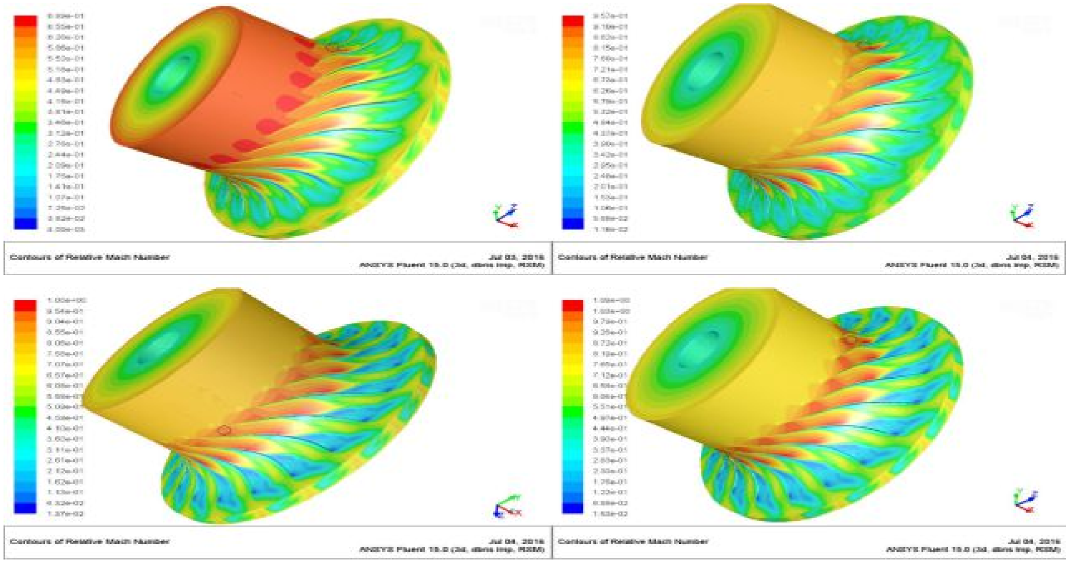
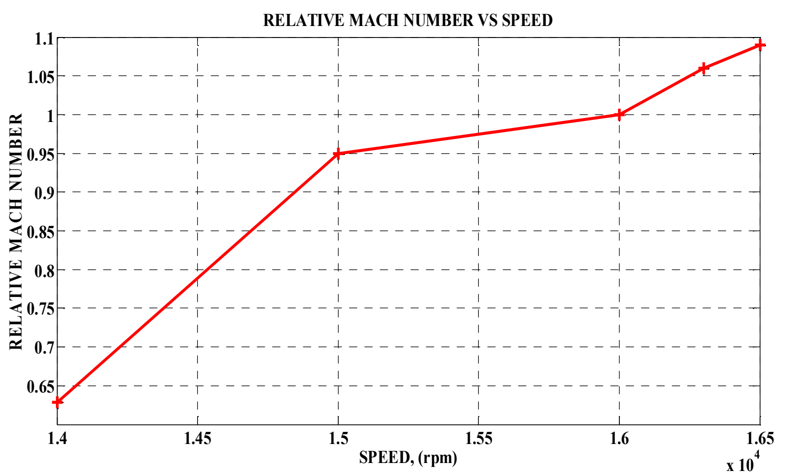


Figure 3 Contours of Relative Mach number at 14000, 15000, 16000, 16500 rpm.



Graph 5 Relative Mach number vs Speed.

Table 6 Speed vs Relative Mach number, Discharge and Pressure ratio.

S.no	Speed	Relative Mach number	Mass flow rate (kg/s)	Pressure ratio (P_d/P_s)
1	14000	0.689	0.27	2.20
2	15000	0.957	0.35	2.39
3	16000	1	0.4	2.64
4	16500	1.09	0.4	2.64

The following conclusions can be drawn from above tables and figures

- Discharge as well as pressure ratio increases with increase in speed and remains constant while relative Mach number has an increasing trend with speed.
- At 16000 rpm the relative velocity has reached sonic velocity and the choking is initiated in the impeller part while at lower speed the relative Mach number is less than one. The discharge has remained constant beyond 16000 rpm and relative Mach number is seen to increase beyond the value of one. It may be noted that the choking condition occurring either in impeller or diffuser depends on the speed while it remains constant and independent of speed when it is initiated in guide vanes (depends on inlet stagnation conditions).

6. CONCLUSION

The Eckardt centrifugal compressor impeller was considered in the present study for investigating the effect of choosing various turbulence models viz Spalart-Allmaras (Curvature Correction),Transition SST (Curvature Correction), Reynolds stress model (Compressibility Effect), Scaled Adaptive Simulations (Curvature Correction + Compressibility Effect) present in fluent software on performance prediction of compressor. It is observed that values of various parameters total pressure, total temperature, blade angles, torque, isentropic and polytrophic efficiencies etc. (except for mass flow) are within range. However due to strong channelled curvature and intensive rotation coupled with high Mach number fluid flow RSM (CE) were considered to be superior for the problem being investigated.

Surge and Choke phenomenon were also investigated in the present study surge generally initiated when discharge becomes almost equal to zero and surging causes severe vibrations coupled with reverse flow. Generally the discharge varies with back pressure at centrifugal outlet. It is observed that discharge has almost decreased to zero value at a back pressure of 1.8 bar indicating onset of surge phenomena at 1.8 bar back pressure, 15000 rpm. Hence to avoid surge it would be necessary to ensure that back pressure doesn’t increase beyond 1.6 bars. Choke phenomena arise when discharge reaches maximum value remains constant when speed is increased further. For the compressor under study the discharge was found to increase from 0.27 kg/s to 0.4 kg/s (per blade) when speed is increased from 14000 rpm to 16000 rpm. The maximum value of 0.4 kg/s remained constant even when speed is increased from 16000 rpm to 16500 rpm indicating onset of choking phenomena at 16000 rpm

REFERENCES

[1] R. Aghaei tog, A.M. Tousi, A. Tourani, “Comparison of turbulence methods in CFD analysis of compressible flows in radial turbomachines”, *Aircraft Engineering and Aerospace Technology: An International Journal* 80/6 (2008) 657–665 © Emerald Group Publishing Limited [ISSN 1748-8842],[DOI 10.1108/00022660810911608]

[2] Reza Aghaei tog and A. Mesgharpoor Tousi, “Design and CFD analysis of centrifugal compressor for a micro gasturbine”, *Aircraft Engineering and Aerospace Technology: An International Journal* 79/2 (2007) 137–143 ©Emerald Group Publishing Limited [ISSN 1748-8842], [DOI 10.1108/00022660710732680].

[3] Michael L. Shur, Michael K. Strelets, and Andrey K. Travin ,Philippe R. Spalart , "Turbulence Modeling in Rotating and Curved Channels: Assessing the Spalart–Shur Correction", Copyright ©2000 by the *American Institute of Aeronautics and Astronautics, AIAA JOURNAL* Vol. 38, No. 5, May 2000.

[4] Pavel E. Smirnov, Florian R. Menter , "Sensitization of the SST Turbulence Model to Rotation and Curvature by Applying the Spalart–Shur Correction Term", *JOURNAL OF TURBOMACHINERY*, Copyright © 2009 by ASME OCTOBER 2009, Vol. 131 / 041010-1, [DOI: 10.1115/1.3070573]

[5] K. J. Elliott, E. Savory, C. Zhang, R. J. Martinuzzi and W. E. Lin, "Analysis of a curvature corrected turbulence model using a 90 degree curved geometry modelled after a centrifugal compressor impeller".

[6] F.R. Menter, M. Kuntz and R. Bender, "A Scale-Adaptive Simulation Model for Turbulent Flow Predictions",Copyright © 2003 by CFX. Published by the *American Institute of aeronautics and Astronautics*, Inc AIAA 2003-767.

[7] Luca Mangani, Ernesto Casartelli, Sebastiano Mauri, "Assessment of Various Turbulence Models in a High Pressure Ratio Centrifugal Compressor with an Object Oriented CFD Code", *JOURNAL OF TURBOMACHINERY*. Copyright © 2012 by ASME NOVEMBER 2012, Vol. 134 / 061033-1

[8] Jie Li, Yuting Yin, Shuqi Li and Jizhong Zhang, "Numerical simulation investigation on centrifugal compressor performance of turbocharger",*Journal of Mechanical Science and Technology* 27 (6) (2013) 1597~1601,DOI 10.1007/s12206-013-0405-3

[9] www.springerlink.com/content/1738-494x.

[10] Shalini Bhardwaj, Dr. K. K.Gupta, "Centrifugal Compressor Analysis by CFD", International Journal of Science and Research (IJSR), Volume 3 Issue 11, November 2014

[11] P. Le Sausse, P. Fabrie, D. Arnou, and F. Clunet, "CFD comparison with centrifugal compressor measurements on a wide operating range", *EPJ Web of Conferences* 45 01059 (2013),DOI: 10.1051/ epjconf / 20134501059, © 2013 by EDP Sciences, 2013

[12] Ing. Martin BABÁK, Ph.D, "CFD analysis of a surge suppression device for high pressure ratio centrifugal compressor", *ANSYS konference* 2010, Frymburk 6. - 8. ríjna 2010

[13] Yu Wang, Zhen Luo, "Simulation and Performance Analysis on Centrifugal Compressors of Different Dimensions and Variable Operation Speed", 978-1-61284-459-6/11/$26.00 © 2011 by *IEEE*.

[14] R. A. Tough, A. M. Tousi, J. Ghaffari, "Improving of the micro-turbine's centrifugal impeller performance by changing the blade angles", Copyright © 2010 *ICCES*, vol.14, no.1, pp.1-22.

[15] G. Sravan Kumar, Dr. D. Azad and K. Mohan Laxmi, Valuation of Turbulence Modelling on Low Speed Centrifugal Compressor Using Computational Fluid Dynamics. *International Journal of Mechanical Engineering and Technology,* 7(6), 2016, pp. 634–641.

[16] S.M. Swamy, V. Panndurangadu and J.M. Shamkumar. Effect of a Tip Clearance on the Performance of a Low Speed Centrifugal Compressor. *International Journal of Mechanical Engineering and Technology*, 8(1), 2017, pp. 178–188.